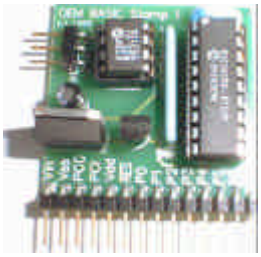
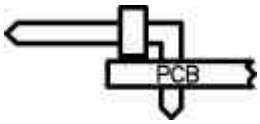
OEM BS1 - Rev A Assembly Instructions



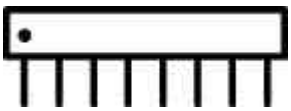
This document is a guide that will aid you in the assembly of your OEM-BS1. It is assumed that you have the proper equipment, and possess the skills necessary to safely assemble electronic components.

Generally, it is best to start with the lower profile components, and then work your way to the tallest components. This is often true because, in tight places, lower profile components are more difficult to get to with taller components in the way.

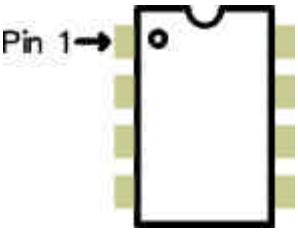
1. The lowest profile components on the board are the right angle strip headers: one should be a 14-position, the other should be 3-position. It may seem ambigious as to which side is inserted into the pcb. The side with the angle should be inserted into the pcb as shown in the drawing to the right. Install both strip headers at this time.



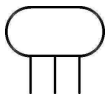
2. The next component to install is the resistor network. A resistor network looks like a comb for a 'Barbie Doll'. On the resistor network (Rnet), there is a dot printed near one end to signify at which end Pin 1 is located. Find Pin 1 on the Rnet and correlate it to the '1' on the PCB that pertains to the 'RN1' part designator. Install the Rnet where designated.



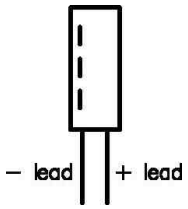
3. The IC sockets are next in line. Nearly all ICs have a 'Pin 1' marker that denotes the location of Pin 1. It is important to install chips with the proper orientation since, more often than not, installing a chip backwards will result in damage to the chip and/or surrounding circuitry. For the IC sockets that you will be installing, there is a notch along the outside edge of the board on one of the short sides. This notch denotes the 'top' of the chip. When oriented properly, Pin 1 is the top-left pin as depicted in the drawing to the right. Install both the 18-Pin and the 8-Pin DIP sockets into U1 and U2 respectively.



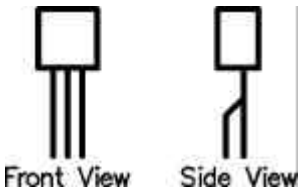
4. The next component to install is the ceramic resonator. The ceramic resonator looks like a coffee bean (although sometimes they are blue instead of brown) with three metal legs sticking out from one side. This is one of the few components that is 'polarity insensitive'. That is to say that it doesn't matter which way it is installed, it will still work OK. Install the ceramic resonator where the 'XTAL' designator is.



5. Next, grab the electrolytic capacitor. It resembles a small cylinder with two long legs emerging from one end. Be particularly careful when installing this component. If it is installed backwards, it can pop like a miniature fire cracker and emit a horrible stench after it is powered up. The electrolytic capacitor will have the negative lead marked with one or more '-' signs. Install the leg NOT marked with the '-' signs in the hole closest to the '+' sign in C1's space on the pcb.



6. Now find the Reset Supervisor; it looks like a black bead with three legs and install it into U3. The white silkscreen on the pcb will guide you as to how to orient this component. You will notice that the middle leg must be bent slightly to accomodate the hole pattern. Please bend the middle leg of the Reset Supervisor according to the side view drawing to the right. Avoid excessive bending as this can damage the component. Install the reset supervisor into U3.



7. The last component to solder on is the voltage regulator, VR1. It is a relatively large plastic square with a metal tab protruding from the top. Install this with the metal tab closest to the electrolytic capacitor, C1. This component, like the electrolytic capacitor, IS polarity sensitive and MUST be installed correctly, lest ye suffer smoke and fire and the gnashing of teeth.

8. Be sure you clean the board before you stuff the Basic Interpreter IC and the eeprom IC into U1's and U2's respective sockets. That's it ! You've done it! You've made your own stamp.

9. Visually inspect the board. Before applying power to it, repair all solder bridges and ensure that all components have been installed in the proper locations and with the correct orientation.

10. Connect the OEM BS1 to a PC parallel port using our standard cable, or make your own based on these connections:

    - OEM BS1 PCI to PC Parallel Port Pin 2 (D0)
    - OEM BS1 PCO to PC Parallel Port Pin 11 (Busy)
    - OEM BS1 Vss to PC Parallel Port Pin 25 (Gnd)

11. Download and run the program, STAMP.exe, from either the CD or the Parallax website. Connect the OEM BS1 to a PC parallel port and power it up. STAMP.exe is a DOS program and must be run on a DOS machine. It is possible, on some machines, to run in a DOS window. To do this, please be sure that your DOS window is running in 'full screen mode', and all other applications are closed. Please consult the Basic Stamp manual, also available from either the CD or our website, for guidance on using your BS1.

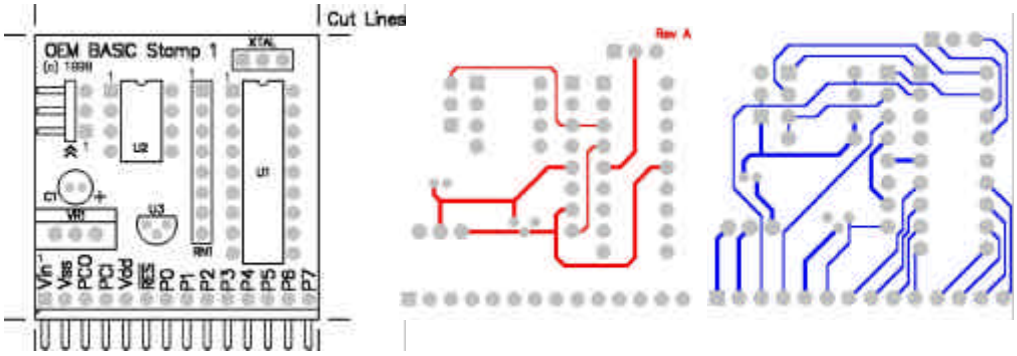
OEM BS1 – Rev A Design Notes

One of the main purposes behind the OEM BS1 is making it easier to embed a BS1 into an OEM design. When designing the BS1 into an your product, please consider the following:

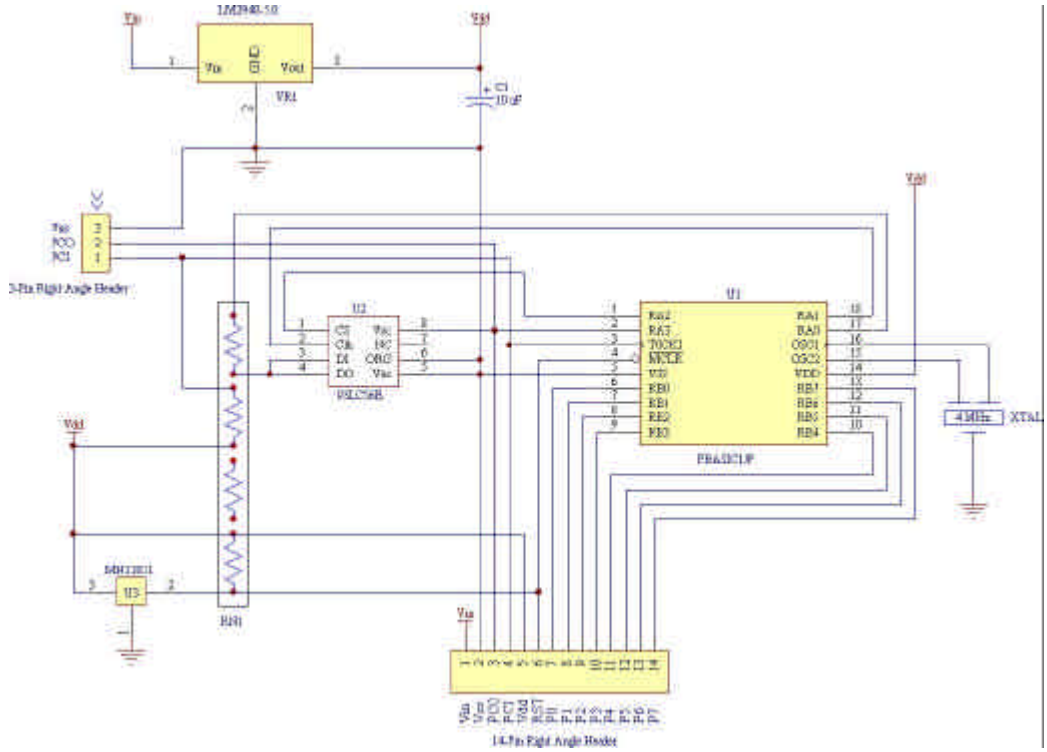
- Power Supply - The Basic Interpreter requires a 'clean' 5 Vdc supply. The LM2940 was selected because of its wide operating range, built-in protection features, and ability to deliver a lot of current. The value of the output capacitor, C1, was based on the manufacturer's recommendation. Be sure to consult the manufacturer's recommended values for input and output capacitance for the regulator that you will be using in your design. Also, if the Basic Interpreter will be located more than a couple of inches away from the power supply, it may be a good idea to add a 0.1 uF ceramic capacitor across Vdd and Vss at the Basic Interpreter IC.

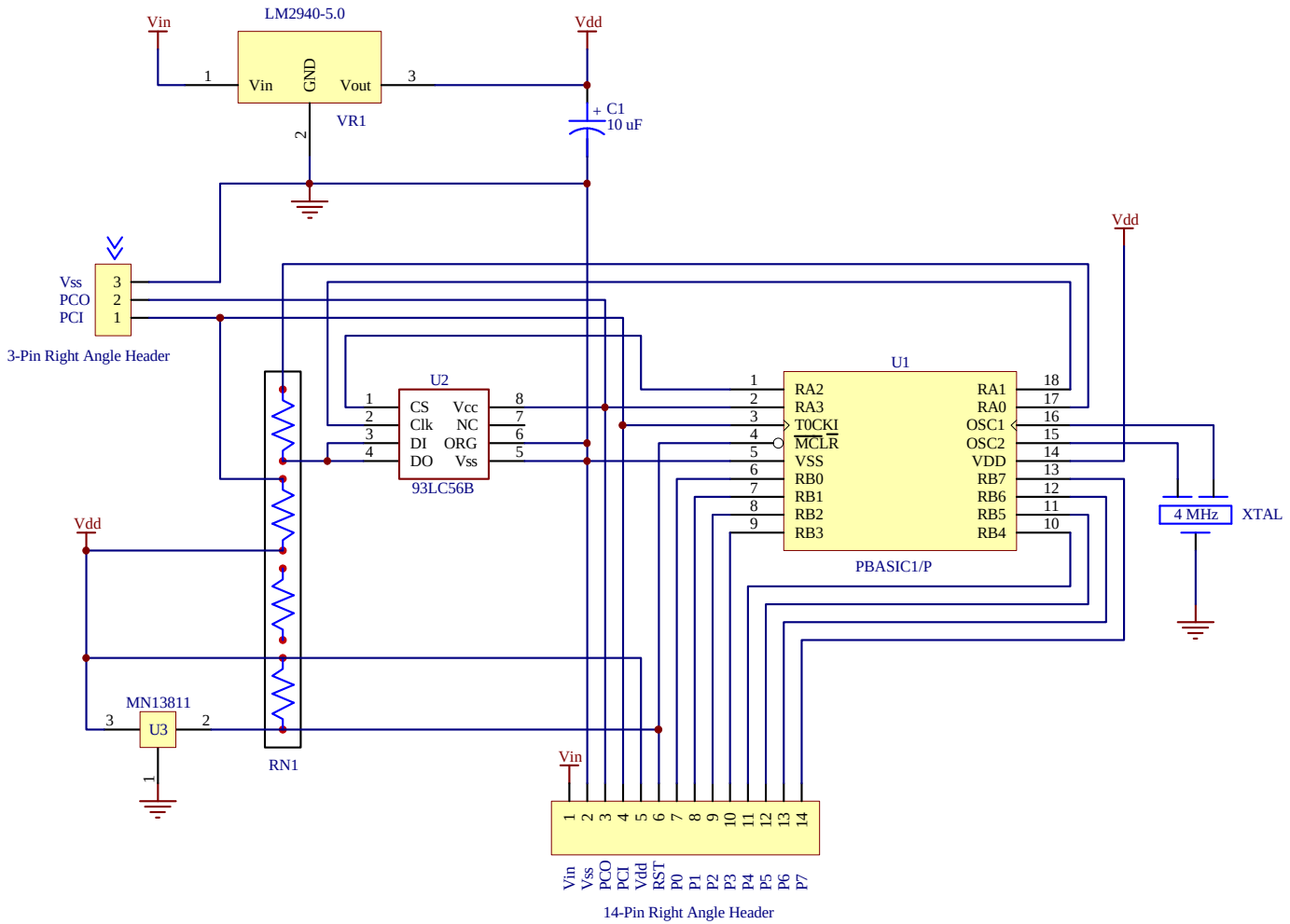
- Resonator Location – Note that the resonator on the OEM BS1 board is located adjacent to the Basic Interpreter. When laying out your board, please choose a location for the resonator that is as close as possible to the Basic Interpreter, and has the shortest circuit traces possible to ensure reliable operation.

- Regulatory Warnings – Stamps and their constituent components are not FCC approved. They are not in finished product form. If you wish to obtain FCC approval, you must first design the stamp into a product, then seek FCC approval of the whole product.



The above images of the board layers are not to scale. Please visit www.parallaxinc.com for the actual source and gerber files. This board was designed using Protel's Client '98 software. The top copper layer is the center plot, and the bottom copper layer is as 'viewed through' the board. Here's the schematic:





Title: OEM Basic Stamp 1		
Size: A	Number:	Revision: A
Date: 2-Feb-2000		Sheet 1 of 1
File: C:\Client98\Sch98\SCHs\OemBS1.sch		Drawn By: John Barrowman